Created in COMSOL Multiphysics 6.1

Flow Around an Inclined NACA 0012 Airfoil

Introduction

This example simulates the flow around an inclined NACA 0012 airfoil using the SST turbulence model and compares the results with the experimental lift data of Ladson (Ref. 1) and pressure data of Gregory and O'Reilly (Ref. 2). The SST model combines the near-wall capabilities of the k-ω model with the superior free-stream behavior of the k-ε model to enable accurate simulations of a wide variety of internal and external flow problems. See the theory for the SST turbulence model in the *CFD Module User's Guide* for further information.

Model Definition

Consider the flow relative to a reference frame fixed on a NACA 0012 airfoil with chord-length $c = 1.8$ m. The temperature of the ambient air is 20 °C and the relative free-stream velocity is $U_\infty = 50$ m/s resulting in a Mach number of 0.15. The Reynolds number based on the chord length is roughly $6 \cdot 10^6$, so you can assume that the boundary layers are turbulent over practically the entire airfoil. The airfoil is inclined at an angle α to the oncoming stream,

$$(u_\infty, v_\infty) = U_\infty(\cos\alpha, \sin\alpha)$$

To obtain a sharp trailing edge, the airfoil is slightly altered from its original shape (Ref. 3),

$$y = \pm c \cdot 0.594689181 \cdot \left(0.298222773 \cdot \sqrt{\frac{x}{c}} - 0.127125232 \cdot \frac{x}{c} - \right.$$
$$\left. 0.357907906 \cdot \left(\frac{x}{c}\right)^2 + 0.291984971 \cdot \left(\frac{x}{c}\right)^3 - 0.105174696 \cdot \left(\frac{x}{c}\right)^4\right)$$

The upstream, top, and bottom edges of the computational domain are located 100 chord-lengths away from the trailing edge of the airfoil and the downstream edge is located 200 chord-lengths away. This is to minimize the effect of the applied boundary conditions.

Figure 1 shows the flow domain and the applied far-field boundary conditions,

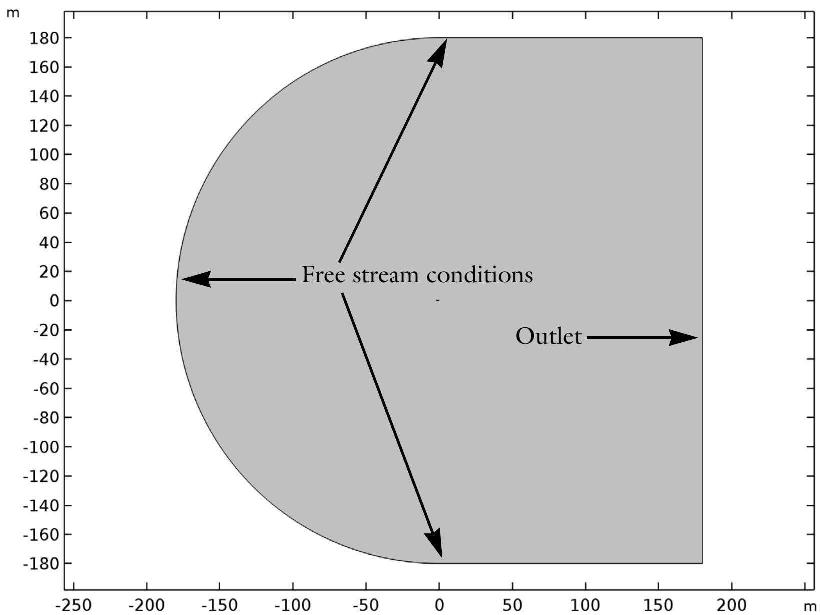


Figure 1: Flow domain and far-field boundary conditions.

Ref. 4 provides far-field values for the turbulence variables,

$$\omega_\infty = (1 \rightarrow 10)\frac{U_\infty}{L}, \qquad \frac{\nu_{T\infty}}{\nu_\infty} = 10^{-(2 \rightarrow 5)}$$

where the free-stream value of the turbulence kinetic energy is given by

$$k_\infty = \nu_{T\infty}\omega_\infty$$

and L is the appropriate length of the computational domain. The current model applies the upper limit of the provided free-stream turbulence values,

$$\omega_\infty = 10\frac{U_\infty}{L}, \qquad k_\infty = 0.1\frac{\nu_\infty U_\infty}{L}$$

Figure 2 shows a close-up of the airfoil section. A no-slip condition is applied on the surface of the airfoil.

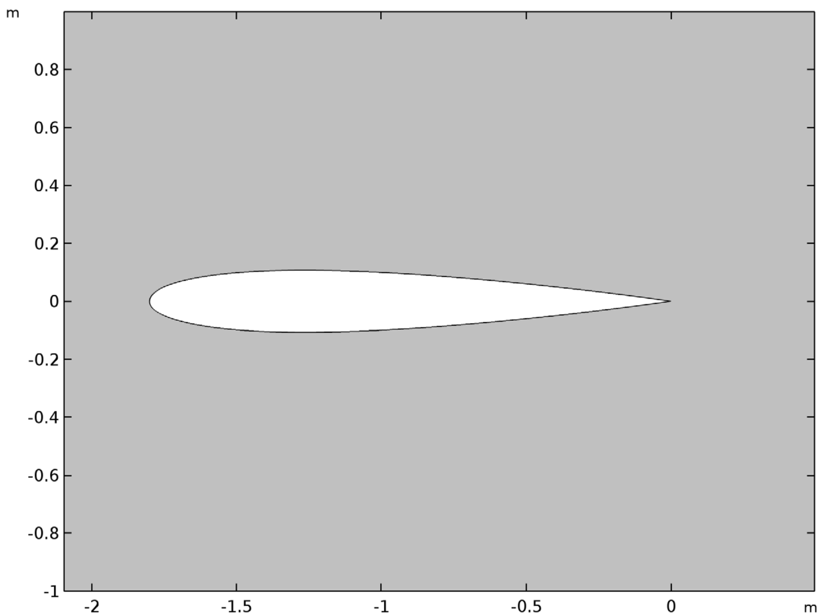


Figure 2: Close-up of the airfoil section.

The computations employ a structured mesh with a high size ratio between the outermost and wall-adjacent elements.

POTENTIAL FLOW SOLUTION

The simplest option when setting the initial velocity field is to use a constant velocity, which does not satisfy the wall boundary conditions. A more accurate and robust initial guess can be obtained solving the potential flow equation.

Assuming irrotational, inviscid flow, the velocity potential φ is defined as

$$\mathbf{u} = -\nabla\varphi$$

The velocity potential φ must satisfy the continuity equation for incompressible flow, $\nabla \cdot \mathbf{u} = 0$. The continuity equation can be expressed as Laplace's equation

$$\nabla \cdot (-\nabla\varphi) = 0$$

which is the potential flow equation.

Once the velocity potential φ is computed, the pressure can be approximated using Bernoulli's equation for steady flows:

$$p = -\frac{\rho}{2}|\nabla\varphi|^2$$

Results and Discussion

The study performs a Parametric Sweep with the angle of attack α taking the values,

$$\alpha = 0°, 2°, 4°, 6°, 8°, 10°, 12°, 14°$$

Figure 3 shows the velocity magnitude and the streamlines for the steady flow around the NACA 0012 profile at $\alpha = 14°$.

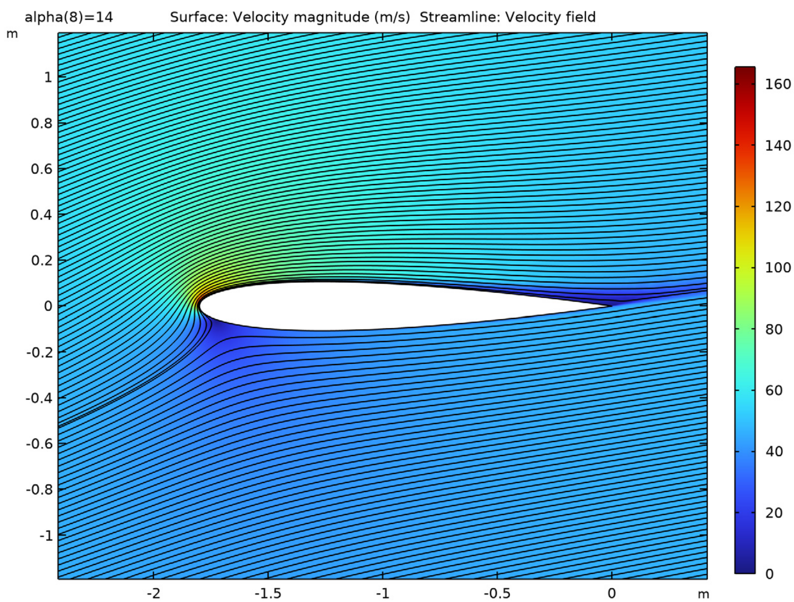


Figure 3: Velocity magnitude and streamlines for the flow around a NACA 0012 airfoil.

A small separation bubble appears at the trailing edge for higher values of α, and the flow is unlikely to remain steady and two-dimensional hereon. Ref. 1 provides experimental data for the lift coefficient versus the angle of attack,

$$C_L(\alpha) = \oint_c (c_p(s)/c)(n_y(s)\cos(\alpha) - n_x(s)\sin(\alpha))\, ds$$

where the pressure coefficient is defined as,

$$c_p(s) = \frac{p(s) - p_\infty}{\frac{1}{2}\rho_\infty U_\infty^2}$$

and c is the chord length. Note that the normal is directed outward from the flow domain (into the airfoil). Figure 4 shows computational and experimental results for the lift coefficient versus angle of attack.

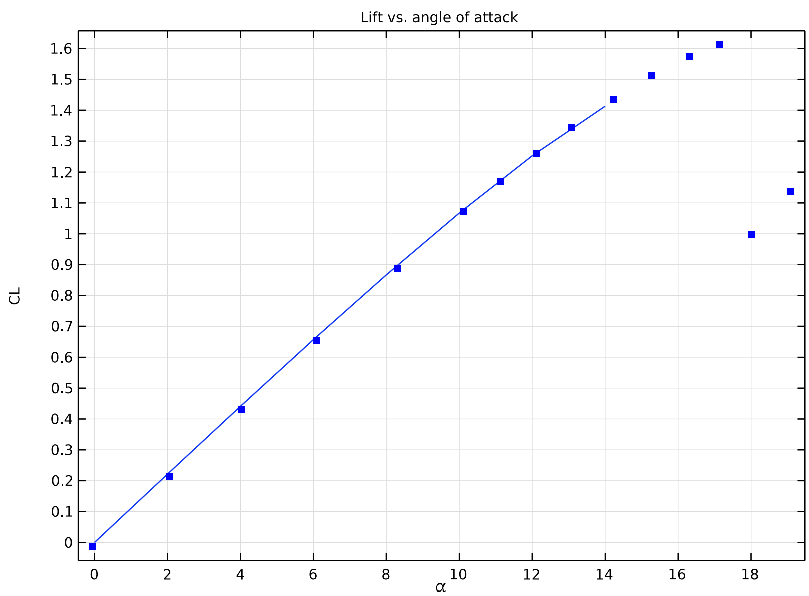


Figure 4: Computational (solid) and experimental (dots) results for the lift coefficient vs. angle of attack.

No discernible discrepancy between the computational and experimental results occurs within the range of α values used in the computations. The experimental results continue through the parameter regime where the airfoil stalls. Figure 5 shows a comparison between the computed pressure coefficient at $\alpha = 10°$ and the experimental results in Ref. 2.

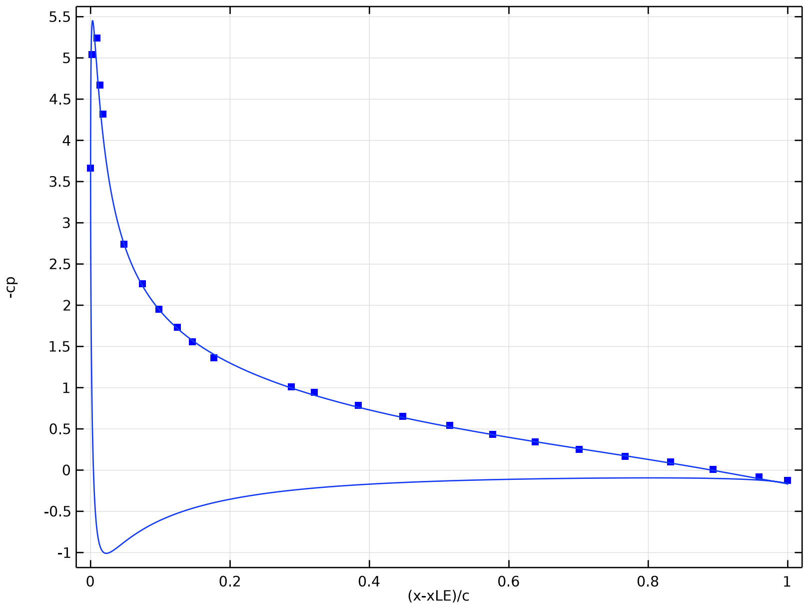


Figure 5: Computational (solid) and experimental (dots) results for the pressure coefficient along the airfoil.

Experimental data is only available on the low-pressure side of the airfoil. The agreement between the computational and experimental results is very good.

Notes About the COMSOL Implementation

The model uses the SST turbulence model together with a Parametric Sweep for the angle of attack to compute the different flows on a mapped mesh.

The initial values for the velocity components are obtained by solving a potential flow equation, which is set up using a PDE interface.

References

1. C.L. Ladson, “Effects of Independent Variation of Mach and Reynolds Numbers on the Low-Speed Aerodynamic Characteristics of the NACA 0012 Airfoil Section,” *NASA TM 4074*, 1988.

2. N. Gregory and C. L. O'Reilly, "Low-Speed Aerodynamic Characteristics of NACA 0012 Aerofoil Section, including the Effects of Upper-Surface Roughness Simulating Hoar Frost," *A.R.C.*, R. & M. no. 3726, 1970.

3. NASA Langley Research Center, Turbulence Modeling Resource, "2D NACA 0012 Airfoil Validation Case," http://turbmodels.larc.nasa.gov/naca0012_val.html

4. F.R. Menter, "Two-Equation Eddy-Viscosity Models for Engineering Applications," *AIAA Journal*, vol. 32, no. 8, pp. 1598–1605, 1994.

Application Library path: `CFD_Module/Verification_Examples/naca0012_airfoil`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Mathematics>Classical PDEs>Laplace's Equation (lpeq)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

LAPLACE'S EQUATION (LPEQ)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laplace's Equation (lpeq)**.

2 In the **Settings** window for **Laplace's Equation**, click to expand the **Dependent Variables** section.

3 In the **Dependent variable** text field, type `phi`.

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Fluid Flow>Single-Phase Flow>Turbulent Flow>Turbulent Flow, SST (spf)**.

4 Click **Add to Component 1** in the window toolbar.

5 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
U_inf	50[m*s^-1]	50 m/s	Free-stream velocity
rho_inf	1.2043[kg*m^-3]	1.2043 kg/m³	Free-stream density
mu_inf	1.81397e-5[kg*m^-1*s^-1]	1.814E-5 kg/(m·s)	Free-stream dynamic viscosity
L	180[m]	180 m	Domain reference length
c	1.8[m]	1.8 m	Chord length
k_inf	0.1*mu_inf*U_inf/(rho_inf*L)	4.184E-7 m²/s²	Free-stream turbulent kinetic energy
om_inf	10*U_inf/L	2.7778 1/s	Free-stream specific dissipation rate
alpha	0	0	Angle of attack

GEOMETRY 1

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type L.

4 In the **Sector angle** text field, type 90.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type 90.

Parametric Curve 1 (pc1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Parametric Curve**.

2 In the **Settings** window for **Parametric Curve**, locate the **Expressions** section.

3 In the **x** text field, type `c*s`.

4 In the **y** text field, type `c*0.594689181*(0.298222773*sqrt(s)-0.127125232*s-0.357907906*s^2+0.291984971*s^3-0.105174696*s^4)`.

5 Locate the **Position** section. In the **x** text field, type `-c`.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 On the object **uni1**, select Domain 2 only.

5 Click **Build All Objects**.

6 Click **Build Selected**.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `L`.

4 In the **Height** text field, type `L`.

5 Click **Build Selected**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

Mirror 1 (mir1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 In the **Settings** window for **Mirror**, locate the **Input** section.

3 Select the **Keep input objects** check box.

4 Click in the **Graphics** window and then press Ctrl+A to select both objects.

5 Locate the **Normal Vector to Line of Reflection** section. In the **x** text field, type `0`.

6 In the **y** text field, type `1`.

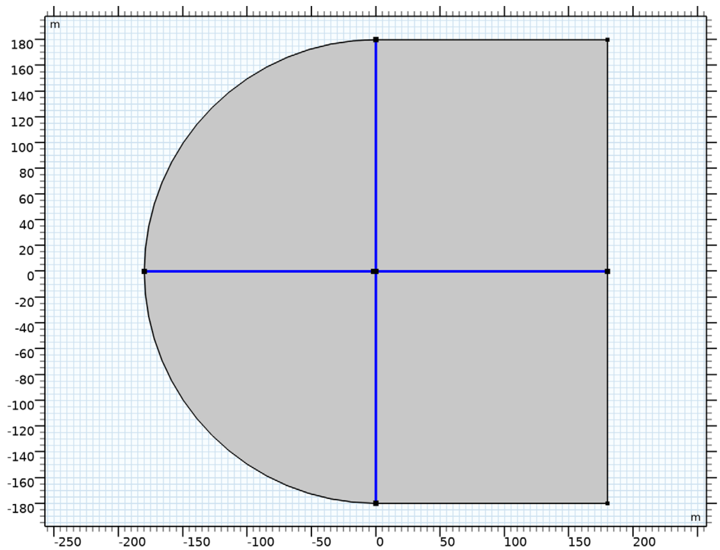
7 Click **Build Selected**.

8 Click the **Zoom Extents** button in the **Graphics** toolbar.

Mesh Control Edges 1 (mce1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Edges**.

2 On the object **fin**, select Boundaries 1, 2, 4, and 5 only.



3 In the **Geometry** toolbar, click **Build All**.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Liquids and Gases>Gases>Air**.

4 Click **Add to Component** in the window toolbar.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

LAPLACE'S EQUATION (LPEQ)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laplace's Equation (lpeq)**.

2 In the **Settings** window for **Laplace's Equation**, locate the **Units** section.

3 Click **Define Dependent Variable Unit**.

4 In the **Dependent variable quantity** table, enter the following settings:

Dependent variable quantity	Unit
Custom unit	m^2/s

5 Click **Define Source Term Unit**.

6 In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	`s^-1`

Dirichlet Boundary Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Dirichlet Boundary Condition**.

2 In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `2` in the **Selection** text field.

5 Click **OK**.

Flux/Source 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Flux/Source**.

2 In the **Settings** window for **Flux/Source**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `1` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Flux/Source**, locate the **Boundary Flux/Source** section.

7 In the g text field, type `-nx*U_inf`.

TURBULENT FLOW, SST (SPF)

Since the density variation is not small, the flow cannot be regarded as incompressible. Therefore set the flow to be compressible.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, SST (spf)**.

2 In the **Settings** window for **Turbulent Flow, SST**, locate the **Physical Model** section.

3 From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.

4 Locate the **Turbulence** section. From the **Wall treatment** list, choose **Low Re**.

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Turbulent Flow, SST (spf)** click **Fluid Properties 1**.

2 In the **Settings** window for **Fluid Properties**, locate the **Distance Equation** section.

3 From the l_{ref} list, choose **Manual**.

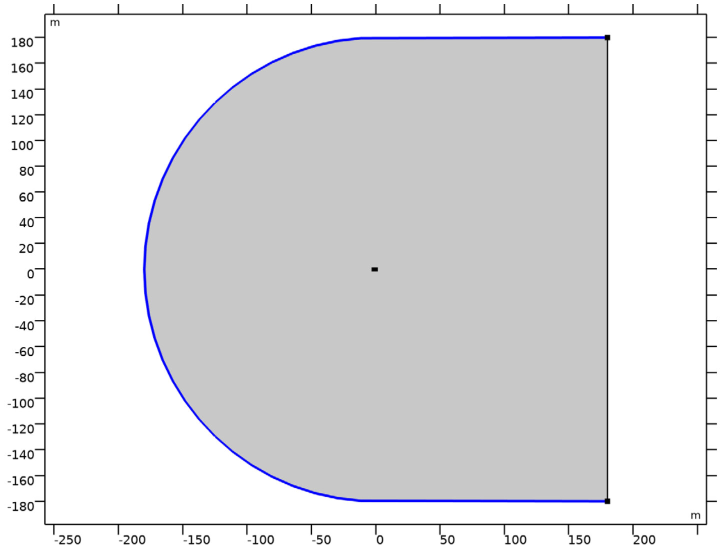
4 In the text field, type `0.2`.

Inlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 1 only.

It might be easier to select the correct boundary by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)



3 In the **Settings** window for **Inlet**, locate the **Turbulence Conditions** section.

4 Click the **Specify turbulence variables** button.

5 In the k_0 text field, type `k_inf`.

6 In the ω_0 text field, type `om_inf`.

7 Locate the **Velocity** section. Click the **Velocity field** button.

8 Specify the $\mathbf{u}_0$ vector as

`U_inf*cos(alpha*pi/180)`	x
`U_inf*sin(alpha*pi/180)`	y

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 Specify the **u** vector as

`-phix`	x
`-phiy`	y

4 In the p text field, type `-spf.rho/2*(phix^2+phiy^2)`.

5 In the k text field, type `k_inf`.

6 In the om text field, type `om_inf`.

Open Boundary 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Open Boundary**, locate the **Turbulence Conditions** section.

4 Click the **Specify turbulence variables** button.

5 In the k_0 text field, type `k_inf`.

6 In the ω_0 text field, type `om_inf`.

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 3 only.

5 Click to expand the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 11 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `100`.

6 In the **Element ratio** text field, type `15000000`.

7 From the **Growth rate** list, choose **Exponential**.

8 Select the **Reverse direction** check box.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 25.

6 In the **Element ratio** text field, type 25.

7 From the **Growth rate** list, choose **Exponential**.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 12 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 25.

6 In the **Element ratio** text field, type 480000.

7 From the **Growth rate** list, choose **Exponential**.

8 Select the **Reverse direction** check box.

Distribution 4

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 100.

Mapped 2

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

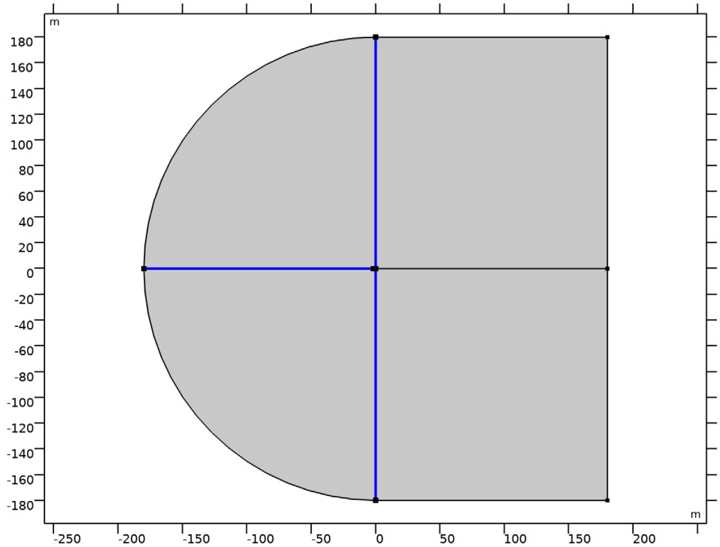
4 Select Domains 1 and 4 only.

5 Locate the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

Distribution 1

1 Right-click **Mapped 2** and choose **Distribution**.

2 Select Boundaries 9–11 only.



3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 100.

6 In the **Element ratio** text field, type 15000000.

7 From the **Growth rate** list, choose **Exponential**.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 2** and choose **Distribution**.

2 Select Boundaries 3 and 4 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 256.

6 In the **Element ratio** text field, type 256.

7 From the **Growth rate** list, choose **Exponential**.

8 Select the **Symmetric distribution** check box.

Mapped 3

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

5 Locate the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

Distribution 1

1 Right-click **Mapped 3** and choose **Distribution**.

2 Select Boundary 10 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 100.

6 In the **Element ratio** text field, type 15000000.

7 From the **Growth rate** list, choose **Exponential**.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 3** and choose **Distribution**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 25.

6 In the **Element ratio** text field, type 25.

7 From the **Growth rate** list, choose **Exponential**.

8 Select the **Reverse direction** check box.

Distribution 3

1 Right-click **Mapped 3** and choose **Distribution**.

2 Select Boundary 8 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 100.

5 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Turbulent Flow, SST (spf)**.

4 In the **Home** toolbar, click **Compute**.

DEFINITIONS

View 1

In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

Axis

1 In the **Model Builder** window, expand the **View 1** node, then click **Axis**.

2 In the **Settings** window for **Axis**, locate the **Axis** section.

3 In the **x minimum** text field, type `-2.5`.

4 In the **x maximum** text field, type `0.5`.

5 In the **y minimum** text field, type `-1.1`.

6 In the **y maximum** text field, type `1.1`.

7 Click **Update**.

View 1

1 In the **Model Builder** window, click **View 1**.

2 In the **Settings** window for **View**, locate the **View** section.

3 Select the **Lock axis** check box.

RESULTS

Potential Flow

1 In the **Model Builder** window, under **Results** click **2D Plot Group 1**.

2 In the **Settings** window for **2D Plot Group**, locate the **Plot Settings** section.

3 From the **View** list, choose **View 1**.

4 In the **Label** text field, type `Potential Flow`.

Surface 1

1 In the **Model Builder** window, expand the **Potential Flow** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `sqrt(phix^2+phiy^2)`.

Streamline 1

1 In the **Model Builder** window, right-click **Potential Flow** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Expression** section.

3 In the **x component** text field, type `-phix`.
4 In the **y component** text field, type `-phiy`.
5 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Starting-point controlled**.
6 From the **Entry method** list, choose **Coordinates**.
7 In the **x** text field, type `0`.
8 In the **y** text field, type `range(-2,0.025,2)`.

Potential Flow

1 In the **Model Builder** window, click **Potential Flow**.
2 In the **Settings** window for **2D Plot Group**, click to expand the **Title** section.
3 From the **Title type** list, choose **Manual**.
4 In the **Title** text area, type `Velocity magnitude and streamlines for potential-flow solution`.
5 In the **Potential Flow** toolbar, click **Plot**.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2 Go to the **Add Study** window.
3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Turbulent Flow, SST>Stationary with Initialization**.
4 Click **Add Study** in the window toolbar.
5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Wall Distance Initialization

1 In the **Settings** window for **Wall Distance Initialization**, locate the **Physics and Variables Selection** section.
2 In the table, clear the **Solve for** check box for **Laplace's Equation (lpeq)**.
3 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
4 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Study** list, choose **Study 1, Stationary**.

Step 2: Stationary

1 In the **Model Builder** window, click **Step 2: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Laplace's Equation (lpeq)**.

4 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Study** list, choose **Study 1, Stationary**.

6 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
alpha (Angle of attack)	`0,2,4,6,8,10,12,14`	

9 In the **Home** toolbar, click **Compute**.

RESULTS

Line Integration 1

1 In the **Results** toolbar, click **More Derived Values** and choose **Integration>Line Integration**.

2 Select Boundaries 3 and 4 only.

3 In the **Settings** window for **Line Integration**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
`p/(1/2*rho_inf*U_inf^2)/c*(spf.nymesh*cos(alpha*pi/180)-spf.nxmesh*sin(alpha*pi/180))`	1	

5 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

6 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

Table 2

1 In the **Results** toolbar, click **Table**.

2 In the **Settings** window for **Table**, locate the **Data** section.

3 Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `naca0012_airfoil_Ladson_CL.dat`.

Table Graph 2

1 In the **Model Builder** window, right-click **1D Plot Group 5** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Table** list, choose **Table 2**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 From the **Color** list, choose **Blue**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

1D Plot Group 5

1 In the **Model Builder** window, click **1D Plot Group 5**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Lift vs. angle of attack`.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** check box. In the associated text field, type `\alpha`.

7 Select the **y-axis label** check box. In the associated text field, type `CL`.

8 In the **1D Plot Group 5** toolbar, click **Plot**.

Table 3

1 In the **Results** toolbar, click **Table**.

2 In the **Settings** window for **Table**, locate the **Data** section.

3 Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `naca0012_airfoil_Gregory_OReilly_Cp.dat`.

TABLE

1 Go to the **Table** window.
2 Click **Table Graph** in the window toolbar.

RESULTS

Table Graph 1

1 In the **Model Builder** window, under **Results>1D Plot Group 6** click **Table Graph 1**.
2 In the **Settings** window for **Table Graph**, locate the **Coloring and Style** section.
3 Find the **Line style** subsection. From the **Line** list, choose **None**.
4 From the **Color** list, choose **Blue**.
5 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

Line Graph 1

1 In the **Model Builder** window, right-click **1D Plot Group 6** and choose **Line Graph**.
2 In the **Settings** window for **Line Graph**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.
4 From the **Parameter selection (alpha)** list, choose **From list**.
5 In the **Parameter values (alpha)** list, select **10**.
6 Locate the **Selection** section. Click **Paste Selection**.
7 In the **Paste Selection** dialog box, type `3 4` in the **Selection** text field.
8 Click **OK**.
9 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
10 In the **Expression** text field, type `-p/(1/2*rho_inf*U_inf^2)`.
11 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
12 In the **Expression** text field, type `(x+c)/c`.
13 In the **1D Plot Group 6** toolbar, click **Plot**.

1D Plot Group 6

1 In the **Model Builder** window, click **1D Plot Group 6**.
2 In the **Settings** window for **1D Plot Group**, locate the **Title** section.
3 From the **Title type** list, choose **None**.
4 Locate the **Plot Settings** section.
5 Select the **x-axis label** check box. In the associated text field, type `(x-xLE)/c`.
6 Select the **y-axis label** check box. In the associated text field, type `-cp`.

7 In the **1D Plot Group 6** toolbar, click **Plot**.

Velocity (spf)

1 In the **Model Builder** window, click **Velocity (spf)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Plot Settings** section.

3 From the **View** list, choose **View 1**.

Streamline 1

Right-click **Velocity (spf)** and choose **Streamline**.

Streamline 1

1 In the **Model Builder** window, expand the **Results>Velocity (spf)** node, then click **Streamline 1**.

2 In the **Settings** window for **Streamline**, locate the **Expression** section.

3 In the **x component** text field, type `u`.

4 In the **y component** text field, type `v`.

5 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Starting-point controlled**.

6 From the **Entry method** list, choose **Coordinates**.

7 In the **x** text field, type `0`.

8 In the **y** text field, type `range(-2,0.025,2)`.

9 In the **Velocity (spf)** toolbar, click **Plot**.